



Aalborg Universitet

**AALBORG UNIVERSITY**  
DENMARK

## **Air Flow in an Exposition Pavilion Studied by Scale-Model Experiments and Computational Fluid Dynamics**

Nielsen, Peter V.

*Publication date:*  
1995

*Document Version*  
Accepted author manuscript, peer reviewed version

[Link to publication from Aalborg University](#)

*Citation for published version (APA):*  
Nielsen, P. V. (1995). *Air Flow in an Exposition Pavilion Studied by Scale-Model Experiments and Computational Fluid Dynamics*. Dept. of Building Technology and Structural Engineering, Aalborg University. Indoor Environmental Engineering No. 27

### **General rights**

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal -

### **Take down policy**

If you believe that this document breaches copyright please contact us at [vbn@aub.aau.dk](mailto:vbn@aub.aau.dk) providing details, and we will remove access to the work immediately and investigate your claim.

# **AIR FLOW IN AN EXPOSITION PAVILION STUDIED BY SCALE-MODEL EXPERIMENTS AND COMPUTATIONAL FLUID DYNAMICS**

**by Peter V. Nielsen, Ph.D., Member ASHRAE**

## **ABSTRACT**

The ventilation design concept, model experiment results, two-dimensional computational fluid dynamics simulation and on-site measurements are presented for the Danish Pavilion project at the 1992 World Exhibition in Seville.

The paper gives a short project history for the building and the ventilation system. The air conditioned restaurant and exhibition hall have a floor area of 450 m<sup>2</sup> (4850 ft<sup>2</sup>) and a mean height of 18 m (59 ft). It is ventilated by a "cooling wall" with a height of 12 m (39 ft). The flow from the cooling wall is similar to the flow in a room with displacement ventilation. Scale-model experiments and computational fluid dynamics simulations indicate a velocity level in the occupied zone of  $\sim 0.6$  m/s ( $\sim 120$  fpm) and the results are confirmed by on-site measurement in the exhibition hall. It is shown that even a simplified two-dimensional flow simulation will give valuable information to be used in the design procedure.

## **INTRODUCTION**

Large ventilated air spaces as shopping arcades, atria and exhibition buildings have become popular in the last decade. The main purpose of designing the air distribution in such constructions is to obtain control of the energy flow and the temperature level. It is very important to have a high ventilation efficiency in the occupied areas and a system which can handle this area without too large air exchange in the rest of the air volume. Smoke movements in case of a fire and necessary escape routes are other important subjects. It is also necessary to limit the air velocity in the occupied areas because people may work at a restricted activity level in the shops and in open offices.

It is not possible to use full-scale experiments in the design of the air distribution system due to large dimensions. It may also be difficult to use simplified design methods as those based on throws of jets and penetration depths of non-isothermal jets. The cause of this is the complicated geometry which is present in some situations. Several sources of the air movement as for example diffusers, pressure difference around the building, cold downdraught and thermal plumes make it also difficult to use simplified methods.

Scale-Model Experiments (SME) and Computational Fluid Dynamics (CFD) are two possible methods for the determination of the air distribution system. They are discussed in connection with the design of an air distribution system for the exhibition hall in the Danish World Exposition Pavilion. The results are further compared with on-site measurements of air velocities obtained in the building by full-scale tests.

Peter V. Nielsen is a Professor at the Department of Building Technology and Structural Engineering, Aalborg University, Sohngårdsholmsvej 57, Aalborg, Denmark.

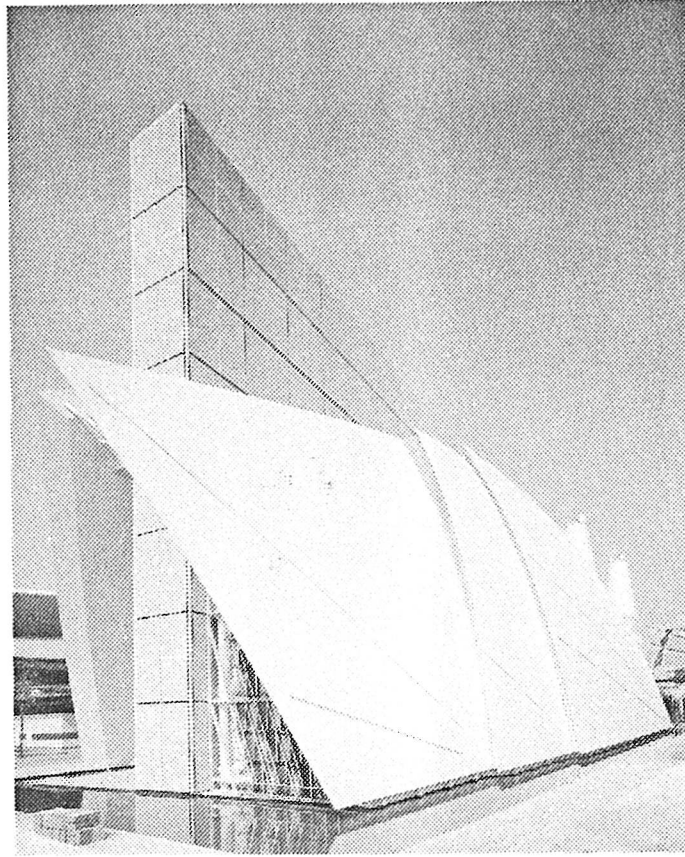


Figure 1. The Danish Pavilion at the World Exposition EXPO'92 in Seville.

The Danish pavilion concept was developed in 1989 and entered the design phase in January 1990. In August 1991 the complete prefabricated building was shipped from Denmark to Spain. The erection in Seville was complete in November 1991 and in March 1992 all services were functioning. On April 20, 1992, the World Exposition "EXPO '92" opened and ran to October 12, 1992, see figure 1. The Danish Pavilion was later bought by Tamba State in Japan and dismantling of the pavilion was completed in December 1992. The pavilion arrived in Japan in February 1993. A further description is given in "The Danish Pavilion" (1992).

## THE BUILDING AND THE VENTILATION SYSTEM

The building has two main elements: a steel framed "container" structure, facing west, with a floor area of  $45.0 \text{ m} \times 2.5 \text{ m}$  ( $147 \text{ ft} \times 8.2 \text{ ft}$ ), a height of 24 m (79 ft) and a fibre glass "sail" construction, facing east, which leans against the container structure, see figure 1. The exhibition room is thus formed between the "sails" and the "containers" and it is enclosed by glass walls to the north as well as to the south. It has an average height of 18 m (59 ft), a length of 45 m (147 ft), an average width of 10 m (33 ft) at floor level and, furthermore, it has a triangular cross section.

It was very quickly determined that a traditional air conditioning system for this exhibition room could neither be accepted architecturally nor economically. Numerous alternatives and ideas were investigated and the nature of the pavilion and the concept of a World Exposition encouraged the pursuit of experiments. The selected design was based simply on installation of exposed cooling elements in the south gable of the room and an extract fan in the top north end of the room, see figure 2. Air is drawn through the cooling elements where it cools down and flows through the occupied zone (Fox and Nielsen 1993).

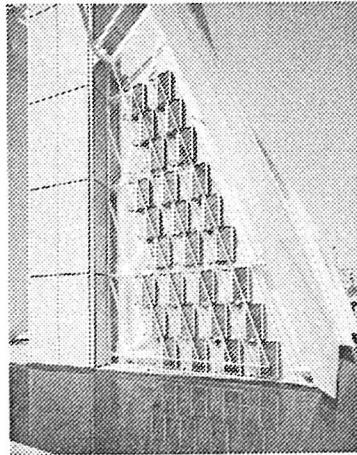


Figure 2. Cooling elements in the south gable of the pavilion.

The design load of the occupied zone in the restaurant and exhibition hall is 48 kW (164000 Btu/h) corresponding to 300 persons in the pavilion. Equipment for slide and video is installed above the occupied zone. This equipment will generate another 130 kW (444000 Btu/h) which is expected to move upward in convective flows and thus causing a high temperature in the upper part of the pavilion. This part of the heat load will be extracted directly by the upper ventilator at a high temperature effectiveness.

The heat load transmission from outside will be small because one side of the exhibition hall is an air conditioned building and the other side is a fibre glass sail with external water cooling. The north wall is a glass window and the south wall - where the solar radiation is present - is the supply wall with cooling devices.

The cooling devices in the south gable are designed to give an inlet air temperature of 26°C (79°F) at an outdoor temperature of 38°C (100°F). The exhaust fan in the ceiling, which is a part of the fire ventilation, is designed for a flow rate of 10 m<sup>3</sup>/s (21000 cfm) corresponding to a temperature difference between return and supply of 4 K (7°F) due to the load in the occupied zone. The load from slide equipment and video will increase this temperature difference.

It is important to restrict the air velocities in the occupied zone. A velocity up to 1 m/s (200 fpm) is acceptable instead of the normal limiting figure of 0.15 m/s (30 fpm) due to the high air temperature in the hall. The conditions are in good agreement with the thermal comfort conditions for 0.3 clo and 1 met, see (Fanger 1967).



## SCALE-MODEL EXPERIMENTS

The following chapter will discuss the theory behind scale-model experiments and show the results obtained from the experiments.

### Dimensionless numbers and similarity principle

It can be shown from theory that a similar flow will be obtained in full-size room and in scale model if the dimensionless boundary conditions are identical and if the Archimedes number, the Reynolds number and the Prandtl number are the same in both situations.

It is impossible to make a scale-model experiment in a strongly reduced scale if all the dimensionless numbers have to be kept constant. If e.g. the scale is reduced by a factor of 10, then the velocity has to be increased by a factor of 10 due to the Reynolds number which will give an increase in the temperature difference by a factor of 1000 in order to keep the Archimedes number. The Prandtl number is, on the other hand, unchanged when air is used as the fluid in the model experiments.

The problem can in practice be overcome if the Reynolds number is high and the flow pattern is governed mainly by fully developed turbulence. It is possible to ignore the Reynolds number and the Prandtl number at a sufficiently high level of velocity because the structure of the turbulence and the flow pattern will be similar at different supply velocities and therefore independent of the Reynolds number. The transport of thermal energy by turbulent eddies will also dominate the molecular diffusion and will therefore be independent of the Prandtl number. See e.g. Nielsen (1993) for a further discussion of scale-model experiments.

### Measurements

The model experiments for the restaurant and exhibition hall were conducted in a model with the scale 1/10. The model room was made with the correct geometrical shapes with two non-parallel sidewalls which gave a triangular vertical cross section perpendicular to the main flow direction and an increased width of the occupied zone in the downstream direction of the flow.

The supply wall with the cooling elements was simulated by a diffuser which covered the whole supply area. The velocities were measured by spherical omnidirectional probes specially developed for low velocity applications. The probes had a measuring accuracy of  $\pm 0.02$  m/s ( $\pm 4$  fpm) and only mean velocities were measured.

Sixteen different experiments were carried out and three of the more essential ones will be discussed in detail in the following chapter. Table 1 shows the main parameters.

In the experiment the Archimedes number is based on the height  $h$  of the inlet device as the length scale and the face velocity  $u_f$  as reference velocity where face velocity is the supply flow divided by a reference area. The experiments do only consider the heat load in the occupied zone.

Table 1. Conditions and results for three model experiments.

	$h$ $m$ ( $ft$ )	$\Delta T_o$ $K$ ( $^{\circ}F$ )	$u_f$ $m/s$ ( $fpm$ )	$Ar$
Test A	1.2 (3.9)	5.6 (10.1)	0.26 (51.1)	3.35
Test B	1.2 (3.9)	6.0 (10.8)	0.175 (34.4)	7.87
Test C	1.2 (3.9)	7.5 (13.5)	0.175 (34.4)	9.84

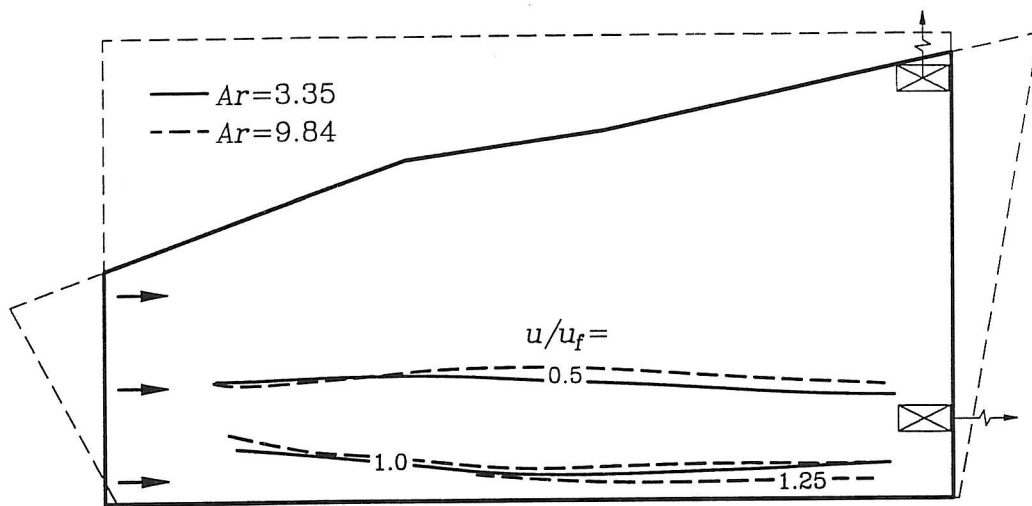


Figure 3. Normalized velocity distribution versus Archimedes number in the exhibition hall.

Figure 3 shows the normalized velocity distribution in the room as a function of the Archimedes number. It is quite obvious that the flow is a stratified flow with the highest velocity in the occupied zone. Smoke measurements show that the cold air from the cooling device moves horizontally along the floor in the restaurant and exhibition section.

Measurements show that the velocity has a fairly constant level in the occupied zone, even far downstream from the wall with the cooling device as shown in figure 3. The flow is plane and it is a general experience that the velocity in a plane stratified flow is constant and independent of distance from the inlet device (Nielsen 1994).

Figure 4 shows the maximum velocity in the stratified flow. The velocity increases for increasing Archimedes number corresponding to a decreasing thickness of the flow. The level of the maximum velocity  $u_m/u_f$  in the occupied zone is 1.1 in test A, 1.5 in test B and 1.8 in test C. The measured velocity close to the supply opening may be slightly influenced by the actual diffuser used in the scale model.

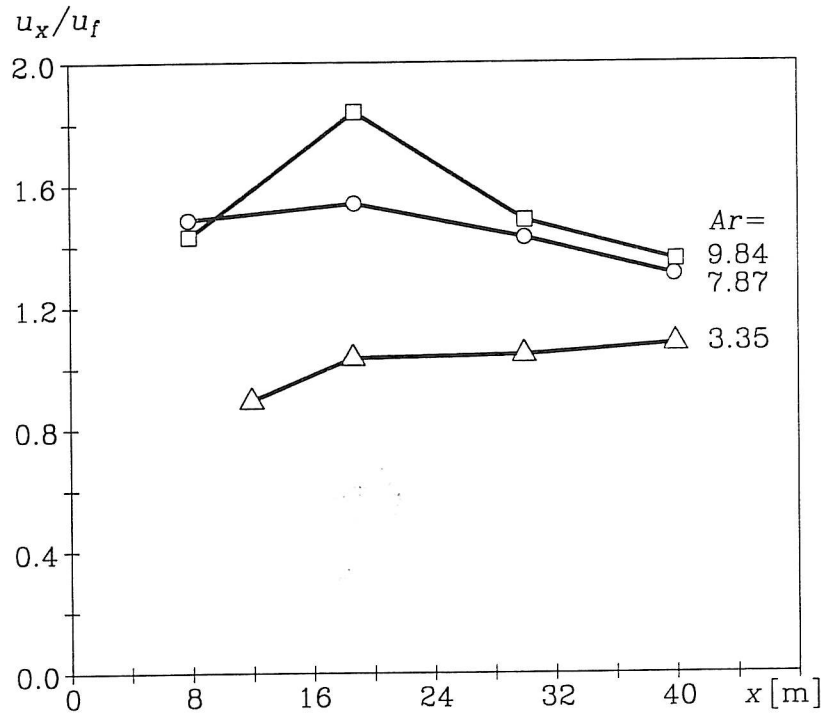


Figure 4. Maximum velocity in the stratified flow versus distance from the supply wall.

The influence from the location of the return opening was tested by using the lower opening alone, the upper opening alone and a combination of both openings with 50% flow in each opening. It was not possible to measure any effect on the velocity level in the model. Experiments with a variable temperature at the surface of the supply device with the highest temperature in the upper part of the device did not show any pronounced influence on the velocity level.

### Transformation of model experiments into full scale

The volume flow rate from the cooling device in the south wall is designed to be 10 m<sup>3</sup>/s (21000 cfm). This is equivalent to a face velocity  $u_f$  of 0.35 m/s (69 fpm). The Archimedes number in test A corresponds to a temperature difference between the return and the supply device of 1 K (1.8°F), corresponding to a heat load of 12 kW (41000 Btu/h) in the restaurant and exhibition hall. This is a rather restricted heat load and the model experiments show that the maximum velocity in the occupied zone will be 0.38 m/s (75 fpm) in full scale. A heavier load of 36 kW (123000 Btu/h) - corresponding to test C - will give up to 0.63 m/s (124 fpm) in the occupied zone. An increase of the flow rate to 14.8 m<sup>3</sup>/s (31400 cfm) will make it possible to handle a load of 40 kW (137000 Btu/h) in the occupied zone with a maximum velocity of 0.57 m/s (112 fpm) as shown in table 2.

Table 2. Transformation of scale-model experiments into full-scale conditions.

	$q_o$ $m^3/s$ ( <i>cfm</i> )	$\Delta T_o$ $K$ ( $^{\circ}F$ )	$u_f$ $m/s$ ( <i>fpm</i> )	$u_m$ $m/s$ ( <i>fpm</i> )	$Q$ $W/m^2$ ( <i>Btu/hft</i> <sup>2</sup> )
Test A	10.0 (21000)	1.0 (1.80)	0.35 (68.9)	0.38 (75)	27.4 (8.70)
Test C	10.0 (21000)	3.0 (5.40)	0.35 (68.9)	0.63 (124)	82.2 (26.10)
Test A	14.8 (31400)	2.2 (3.96)	0.52 (102.4)	0.57 (112)	91.3 (29.00)

## COMPUTATIONAL FLUID DYNAMICS

The fluid dynamics research is strongly influenced by the increasing computer power which has been available for the last decades. This development has the effect that the cost for a given job will decrease by a factor of 10 during every eighth year. The development shows not only a decreasing cost but the computer time is also decreasing. There are more reasons for this development. Firstly, the computer efficiency is increasing more rapidly than the computer costs and this tendency seems to continue. Secondly, a process takes place which increases the flexibility of different software as pre- and post-processor software and, furthermore, there is a continuous development of new software. Improvements in the fundamental routines as e.g. the grid generation procedure and the numerical method do also contribute to an increasing speed.

These tendencies have also influenced the indoor environmental technology. Today a large amount of codes developed at the Universities and commercial developed codes is available for the prediction of air distribution in rooms. This chapter will show the predictions obtained by a simple two-dimensional code to give an example of the information which can be obtained by using such a method compared with scale-model experiments.

A general review of CFD-applications within the indoor environmental technology is given in "Building Systems: Room Air and Air Contaminant Distribution" (Christianson 1989), and current status and capabilities are addressed by Jones and Whittle (1992). The prospects for CFD in room air distribution control have recently been reviewed by Nielsen (1994A).

### Governing equations and numerical method

The air movement in a room with recirculation flow and temperature differences is described by the transport equations for mass, momentum and energy. It is widely used to describe the turbulence by the eddy viscosity concept. The eddy viscosity  $\mu_t$  can be predicted from a  $k$ - $\epsilon$  turbulence model consisting of two transport equations for turbulent kinetic energy  $k$  and dissipation of turbulent kinetic energy  $\epsilon$ , see (Launder, Spalding and Whitelaw 1973).

The  $k-\epsilon$  model is only valid for a fully developed turbulent flow. This is supposed to be the case in this situation with very large dimensions as discussed in the last chapter on scale-model experiments. The CFD-method can be used to make a review of the level of the turbulence. This will be discussed in connection with the appendix.

Low turbulence effects can be predicted in near wall regions with a low Reynolds number model (LRN) as given by Launder and Sharma (1978). However, the model is not suitable in a general form for prediction of turbulence far from the surfaces.

The transport equations are reformulated into finite difference equations and solved by an iteration procedure. A detailed description of the method and especially of the finite volume concept is given by Patankar (1980). Baker, Williams and Kelso (1993) show the application of a finite element method for the prediction of room air distribution.

### Boundary conditions

It was decided to make the CFD-predictions in a two-dimensional Cartesian code to study the level of information which can be obtained in this simplified situation in comparison with scale-model experiments. The code is based on time averaged transport equations for momentum, mass and energy and a  $k-\epsilon$  turbulence model. The numerical method uses finite volume concept and staggered grid.

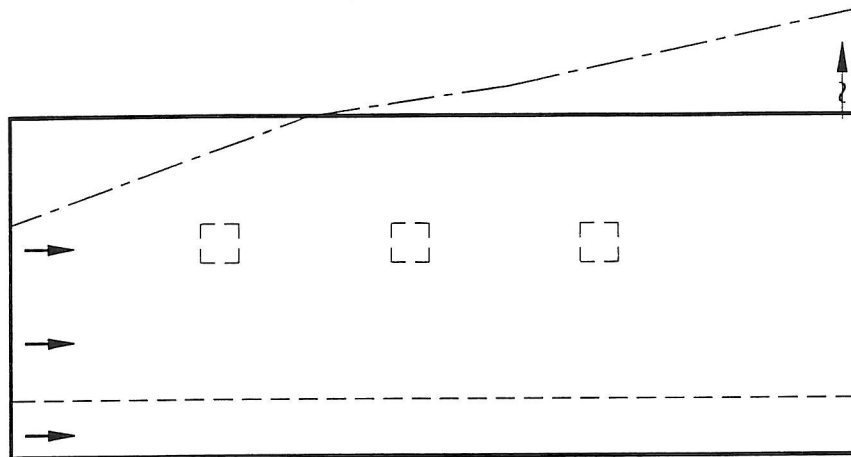


Figure 5. Boundary conditions for two-dimensional CFD-predictions. The dotted lines show the location of volume heat sources.

Figure 5 shows the boundary conditions for the geometry. The room has the height of 18 m (59 ft) and 12 m (39 ft) of the south wall are covered by the supply device. The return opening is located in the north corner of the ceiling. The real exhibition hall has a triangular shape perpendicular to the direction of the main flow. This is impossible to take into consideration in this case with a two-dimensional configuration of the CFD-method. It is also obvious from figure 5 that the geometry of the ceiling regions is simplified in the CFD-predictions.

The locations of the heat sources are shown in figure 5. The heat load from the occupied zone is described as a volume source with a height of 3 m (10 ft). One prediction (case D) does

also take account of additional heat sources with a higher location as shown by means of the three volumes in the figure.

Table 3. Boundary conditions for the CFD-predictions.

	$h$ $m$ ( $ft$ )	$u_f$ $m/s$ ( $fpm$ )	$Q$ $W/m^2$ ( $Btu/hft^2$ )	$Ar$ Full scale	$Ar_{2D}$ 2D Prediction
Case A	12 (39)	0.35 (68.9)	27.4 (8.70)	3.35	0,78
Case C	12 (39)	0.35 (68.9)	82.2 (26.10)	9.84	2.35
Case D	12 (39)	0.35 (68.9)	27.4 + 296.7 (8.70 + 94.20)		9.29

The two-dimensional predictions are made for full-scale situations corresponding to test A and test B in order to facilitate comparisons with the scale-model tests. Table 3 shows that the face velocity  $u_f$  is used for the description of the mass flow rate, which means that the flow rate per supply area is identical in predictions and in full scale. The heat load per floor area is also identical in predictions and in full-scale. The assumptions will have the consequences that the Archimedes numbers in full scale flow are different from the Archimedes numbers in the two-dimensional predictions because the effective supply areas are slightly different and, especially, because the real hall has a triangular shape which corresponds to a low "velocity" in the Archimedes number. It is important to work with the correct flow rates of energy and momentum and therefore it is advisable to use the boundary conditions in table 3 when the results are compared with the measured flow fields in test A and test C.

## Predictions

The predicted flow field is unidirectional in the main part of the hall including the occupied zone and it has a small recirculation above the supply opening. The maximum velocity is located at the height of 0.5 to 1.0 m (1.6 to 3.3 ft) which is similar to the relative location measured in the model experiments.

Figure 6 shows the maximum velocity in the stratified flow versus distance from the south wall with the supply opening. The velocity level for the two Archimedes numbers is almost identical to the velocities measured in the scale model, see figure 4. The scale-model experiments obtain the highest velocities close to the supply opening while the CFD- predictions have the highest velocities in the opposite part of the hall. This may partly be explained by the fact that the real hall is narrow close to the supply wall ( $\sim 6.2$  m ( $\sim 20$  ft)) and wide in the other end at the north wall ( $\sim 13.4$  m ( $\sim 44$  ft)) and partly by the fact that the velocities measured in the scale model close to the supply opening may be influenced by local velocity gradients generated by the actual diffuser in the model.



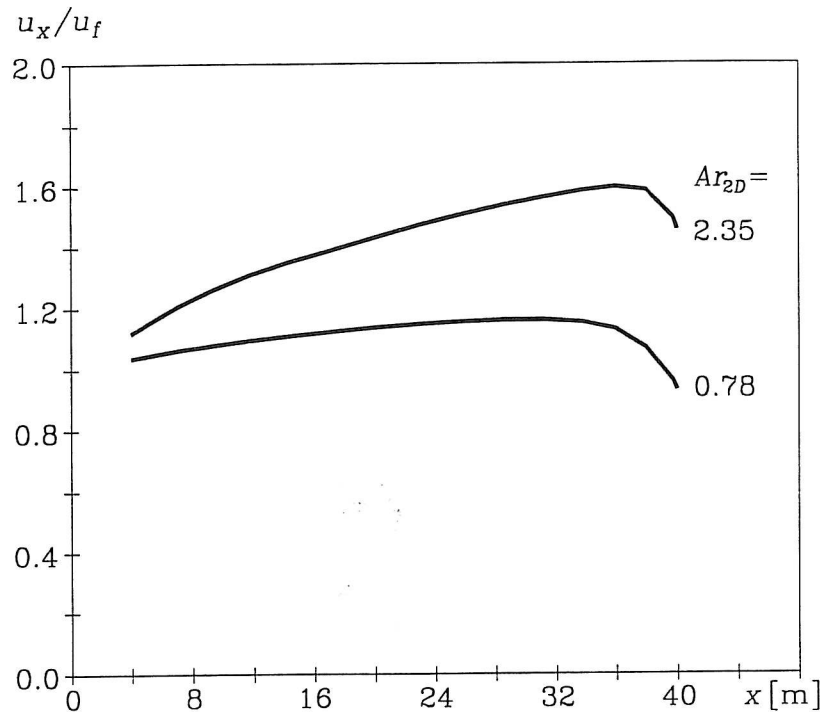


Figure 6. Maximum velocity in the flow versus distance from the supply wall.

The maximum velocity in the occupied zone is 0.41 m/s (81 fpm) in case A and 0.56 m/s (110 fpm) in case B. The full-scale level of the maximum velocity in scale-model test A and test B is 0.38 m/s (75 fpm) and 0.63 m/s (124 fpm), respectively. Comparisons between SME and CFD show an accuracy which is far above an expected level when all the simplifications in the boundary conditions are taken into consideration.

It is generally expected that the flow in the occupied zone is unaffected by high located heat sources. The situation is examined in case D where the equipment for slide and video is simulated as three sources with a total heat flux of 130 kW (444000 Btu/h) situated as indicated in figure 5. Comparisons with case A show that the maximum velocity in the occupied zone has the same level (0.43 m/s (85 fpm)) but the location is now in the south end of the hall.

Figure 7 shows the temperature profiles at  $x = 16$  m (52 ft) which is between two of the heat sources. Both profiles show that the temperature in the occupied zone is uninfluenced by the high located volume sources. The additional heat sources add a stratification to the flow but this does not influence the temperature in the occupied zone. The predictions are made without a radiation from the ceiling. There must be some effect in practice but it is restricted by the external water cooling of the building surface.

The appendix shows that the flow can be expressed as a fully developed turbulent flow. The turbulence is mainly generated at the supply wall and not by velocity gradients in the room. It is therefore the design of the supply devices which dictates the level of the turbulence and it is the dimensions and velocity at the supply wall which give the characteristic Reynolds number.

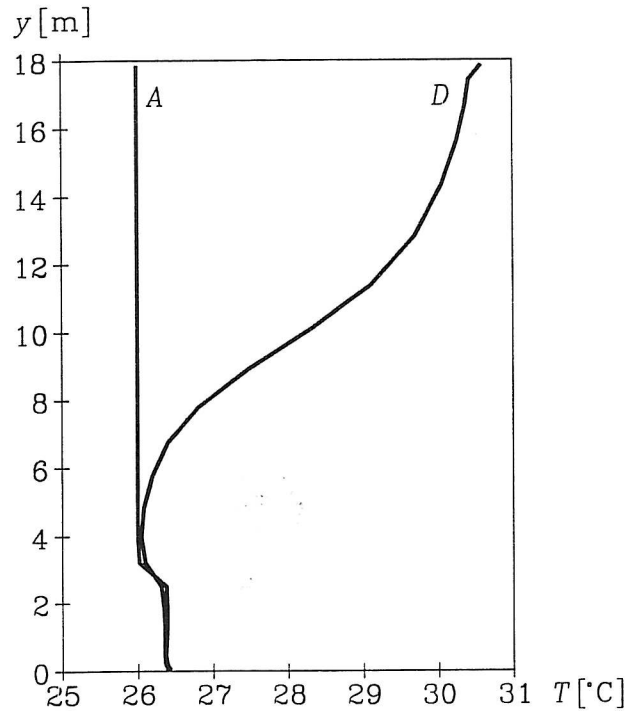


Figure 7. Vertical temperature profiles at  $x = 16$  m (52 ft) for case A and case D.

All the predictions given in this chapter are obtained from a typical commercial code. The specification and computing time are low, partly due to an efficient front end code and partly due to the use of a two-dimensional flow simulation. The boundary conditions correspond to an unstable situation (low location of a heat source and cold flow above) and this affects the stability of the convergence. It is necessary to have some experience in CFD in connection with the use of underrelaxation on the finite difference equations.

## FULL-SCALE ON-SITE MEASUREMENTS

On-site measurements show air velocities at a level of 0.8 m/s (157 fpm) roughly corresponding to the velocity level obtained by the model experiments, see table 4. The velocity has a quicker reduction than expected downstream in the occupied zone, probably due to the influence from the high located heat sources. (This effect was also observed in the case D simulations). The measurements could only take place in the early morning of May 6th, 1992, when the outdoor temperature was 19.7°C (67.5°F) and the inlet air temperature was 12.1°C (53.8°F). Show projectors and lighting were at full power whereas occupant loading was light, approximately 20 people. Discharge air temperature was 27.5°C (81.5°F), indicating the expected strong vertical stratification corresponding to a high ventilation effectiveness. The Archimedes number was  $\sim 50$ , see also Fox and Nielsen (1993).

Table 4. On-site measurements 1 m (3.3 ft) above floor level. Mean temperature and maximum velocity measured across the room versus distance from supply wall.

Distance	Temperature, °C (°F)	Distance	Velocity, m/s (ft)
2 m (6.5 ft)	12.7 (54.9)	2 m (6.5 ft)	0.6 (118)
16 m (52.5 ft)	14.1 (57.4)	3 m (9.8 ft)	0.6 (118)
24 m (78.7 ft)	14.6 (58.3)	5 m (16.4 ft)	0.8 (157)
32 m (105.0 ft)	14.9 (58.8)	8 m (26.2 ft)	0.5 (98)
40 m (131.2 ft)	14.7 (58.5)	12 m (39.4 ft)	0.1 (20)

The conclusions from on-site measurements are: (a) Uniform temperature distribution in the occupied zone with a small horizontal temperature gradient. (b) Strong vertical stratification ensuring separation between occupied zone loads and other room loads. (c) Air velocities equal to those predicted by scale-model experiments and CFD.

## CONCLUSIONS

Scale-Model Experiments (SME) and Computational Fluid Dynamics (CFD) simulations are important design methods.

The air movement in large enclosures can be investigated in a scale model. Special attention must be paid to low-turbulent effect in SME. Large dimensions and large Reynolds number will ensure high-turbulent flow and the Archimedes number is the only important parameter in the experiments.

SME shows that the flow in the exhibition hall is unidirectional and stratified. The experiments indicate a velocity level of  $\sim 0.6$  m/s ( $\sim 120$  fpm) through the whole occupied zone.

CFD-prediction of air movement in large enclosures is a very important alternative to scale-model experiments. The turbulence model in a CFD-method is based on fully turbulent flow (large Reynolds number) and this is easily obtained in a large enclosure.

CFD-predictions confirm the general flow pattern obtained by SME. A velocity level of  $\sim 0.6$  m/s ( $\sim 120$  fpm) is indicated by two-dimensional flow simulation. It is possible to show that the flow in the occupied zone is unaffected of heat sources in the upper part of the hall.

Specification- and computing time are very low for two-dimensional flow simulation. Therefore, it may be very efficient to include two-dimensional flow simulation at an early

stage in the design phase because the predictions are valuable for further design work. Skill and experience are required for the specification work and handling of convergence problems.

On-site measurements in full scale show that the temperature distribution in the occupied zone only have a small horizontal gradient. There is a strong vertical stratification which ensures separation between occupied zone load and other room loads. The air velocity level in full-size measurements is equal to the level obtained by SME and CFD.

## NOMENCLATURE

$Ar$	Archimedes number
$Ar_{2D}$	Archimedes number for 2D predictions
$c_\mu$	Constant in turbulence model
$f_\mu$	Variable in turbulence model
$f_2$	Variable in turbulence model
$h$	Height of supply wall
$k$	Turbulent kinetic energy
$q_o$	Flow rate
$Q$	Heat load per floor area
$R_t$	Local turbulent Reynolds number
$u_o$	Supply velocity
$u_f$	Face velocity
$u_m$	Maximum velocity in the occupied zone
$x$	Horizontal distance from supply wall
$\Delta T_o$	Temperature difference
$\epsilon$	Dissipation of turbulent kinetic energy
$\mu$	Molecular viscosity
$\mu_t$	Turbulent viscosity

## REFERENCES

- Baker, A.J., P.T. Williams and R.M. Kelso (1993). Development and Validation of a Robust CFD Procedure for Predicting Indoor Room Air Motion. Proc. of the 6th International Conference on Indoor Air Quality and Climate, INDOOR AIR'93, Vol. 5 Helsinki, pp. 183-188.
- Christianson, L.L., (ed.), (1989). Building Systems: Room Air and Air Contaminant Distribution. ASHRAE, ISBN 0-910110-64-6.
- The Danish Pavilion - EXPO'92 Seville. (1992). The Danish Architectural Press, ISBN 87 7407 126 2.

Fanger, P.O. (1967). Calculation of Thermal Comfort: Introduction of a Basic Comfort Equation. ASHRAE Transactions, 73, II.

Fox, S.G. and P.V. Nielsen (1993). Model Experiments in 1990 and On-Site Validation in 1992 of the Air Movement in the Danish Pavilion in Seville. Proc. of the 6th International Conference on Indoor Air Quality and Climate, INDOOR AIR'93, Vol. 2, Helsinki, pp. 675-680.

Jones, P.J. and G.E. Whittle (1992). Computational Fluid Dynamics for Building Air Flow Prediction - Current Status and Capabilities, Building and Environment, Vol. 27, No. 3, pp. 321-338.

Launder, B.E. and B.I. Sharma (1978). Letters in Heat and Mass Transfer, 1, 129.

Launder, B.E., D.B. Spalding and J.H. Whitelaw (1973). Turbulence Models and their Experimental Verification. Imperial College, Heat Transfer Section, Reports HTS /73/.

Nielsen, P.V. (1993). Model Experiments for the Determination of Airflow in Large Spaces. Proc. of the International Conference INDOOR AIR'93, Vol. 5, Helsinki, pp. 253-258.

Nielsen, P.V. (1994). Stratified Flow in a Room with Displacement Ventilation and Wall-Mounted Air Terminal Devices. ASHRAE Transaction, V. 100, Pt. 1, pp. 1163 - 1169.

Nielsen, P.V. (1994A). Prospects for Computational Fluid Dynamics in Room Air Contaminant Control. Proceedings of the 4th International Symposium on Ventilation for Contaminant Control, Ventilation'94, Center for Research on Occupational Health, Stockholm, Part 1, pp. 26 - 46.

Patankar, S.V. (1980). Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corporation, Washington.

## APPENDIX

Launder and Sharma (1978) work with a low Reynolds number model (LRN) which among other things contains the coefficients  $f_\mu$  and  $f_2$ . The coefficients make a modification of the turbulence model in case of a low local turbulent Reynolds number  $R_t$

$$f_\mu = \exp(-3.4/(1 + R_t/50)^2) \quad (A1)$$

$$f_2 = 1 - 0.3 \exp(-R_t^2) \quad (A2)$$

A high turbulent  $k$ - $\epsilon$  model, which is used in this paper, may be considered as a special version of a LRN model with  $f_\mu$  and  $f_2$  equal to 1.0. Those conditions are fulfilled in practice when  $R_t$  exceeds 400 everywhere in the flow domain as seen from (A1) and (A2).

The local turbulent Reynolds number can be expressed by

$$R_t = \frac{1}{c_\mu f_\mu} \frac{\mu_t}{\mu} \quad (A3)$$

where  $\mu_t$  is turbulent viscosity and  $\mu$  is the molecular viscosity of the air.  $R_t > 400$  and  $c_\mu = 0.09$  corresponds to

$$\mu_t/\mu > 40 \quad (A4)$$

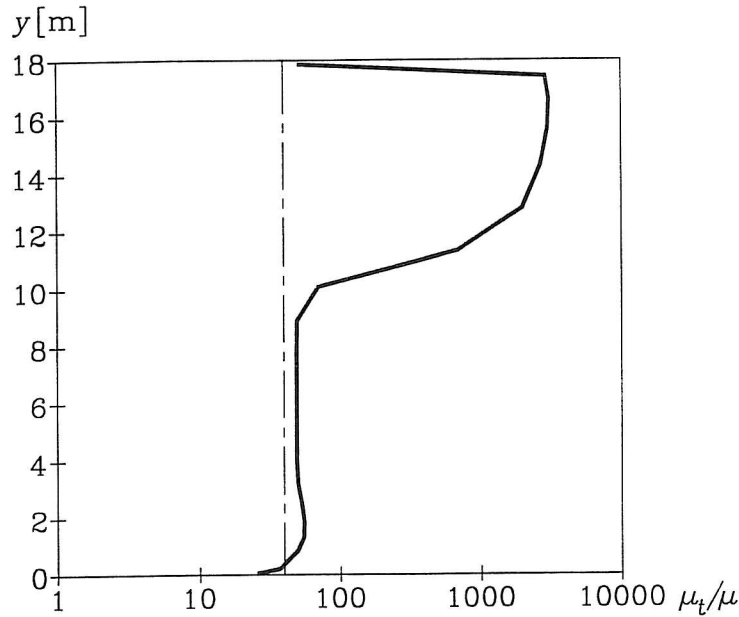


Figure 8. Vertical turbulent viscosity profile at  $x = 16$  m (52 ft) for case A.



Figure 8 shows the distribution of  $\mu_t/\mu$  at  $x = 16$  m (52 ft). Equation (A4) is fulfilled at this vertical section as well as in the rest of the main flow. This may indicate that the flow is fully turbulent which also is a necessary assumption for the predictions because the  $k-\epsilon$  model has been used as a turbulence model. The turbulence is mainly generated at the supply wall where it has a level of  $\mu_t/\mu \sim 50$ . This is a low level taking the large dimensions into consideration and it is much higher in many practical situations. Velocity gradients are one of the important sources to turbulence and the upper part of figure 8 shows an excreased level which can be obtained in areas with recirculation.